Numerical Simulation of Fire in an Aircraft Engine Nacelle

V.F. Nicolette, S.R. Tieszen, and L.A. Gritzo
Sandia National Laboratories[1]
Unsteady and Reactive Fluid Mechanics Dept. 9116
Albuquerque, NM 87185-0836

This paper summarizes the results of a field model simulation of a fire in a geometry representative of an aircraft engine nacelle. Interesting phenomena such as flame stabilization on clutter are predicted. This work provides a basis for the investigation of more detailed flame stabilization studies with the model, and also for nacelle fire suppression studies.

Problem Description

A complex geometry was generated to simulate an aircraft engine nacelle. The geometry generated was not intended to represent a specific nacelle, but rather to be generic in nature. It was developed based on drawings of an actual nacelle, including appropriate dimensions. The annular region simulated is approximately 0.1 m thick, and 1.5 m long, with an outer radius of 0.5 m. This annular region (the nacelle) represents the volume between the combustion chamber and the engine housing. Many complex-shaped objects (referred to as 'clutter') were placed inside the annulus, corresponding to wires, fuel lines, harnesses, valves, etc., that are typically found in nacelles. A three-dimensional finite difference grid of 82 x 34 x 108 cells was used to model the nacelle.

Air injection was prescribed at 4 radial ports near the front end of the nacelle (velocity 28 m/s). Fuel injection was also prescribed near the front end of the nacelle. JP-8 aviation fuel (vapor) was injected at a velocity of 2.8 m/s, simulating a leaking high pressure fuel line. The rear part of the annulus was assigned an outflow boundary condition (constant pressure of 10^5 Pa). Obstacles in the annulus were assigned typical thermal properties so that their thermal response could be included. An initial axial velocity field of 10 m/s was assigned to all of the free flow cells to represent near steady-state flow. There was no delay between the fuel injection process and the prescribed ignition time.

Numerical Model

The VULCAN[2] fire field model (based on the KAMELEON Fire model, Holen, et al., (1990)) was used as a basis for the numerical calculations. The model is 3-dimensional and transient, and uses an extension of the SIMPLEC method. The combustion and soot models of Magnussen are used (Magnussen, et al. (1979) and Magnussen (1981)), along with the discrete transfer radiation method of Shah (1979). The model has both a k-ε turbulence submodel and a k-L LES submodel. The standard k-ε submodel was used for these calculations. Convective heat transfer and momentum transfer to objects were treated using wall functions.

Results and Discussion

Calculations were carried out for 1 second of real time. The fire had reached a steady-state condition by ~0.3 seconds due to the high forced air flow. Complex velocity fields are present in the calculations as a result of the clutter and the fuel and air injection, as indicated by strong vortices throughout the volume.

---

1. Sandia National Laboratories is managed and operated by Lockheed Martin Corporation for the U. S. Department of Energy under contract DE-AC04-94AL85000.
2. VULCAN is under joint development at Sandia National Laboratories and SINTEF/NTH (Norway).

As a result of the complex velocity field and the associated mixing of fuel and air, very complex flame shapes are observed. These appear to be driven by both the air injection and the volumetric expansion of burning fuel. Near the location of fuel injection, a fuel-rich region exists in which no burning occurs. Immediately adjacent to this area of no-burning is a convoluted flame zone, resulting from the mixing of sufficient air into the fuel vapor to bring the local mixture to a near stoichiometric condition. Unlike typical industrial combustion processes (e.g., pre-mixed applications), which are *forced* phenomena, fires represent a *natural* balance point between fuel, oxidizer, and energy. As a result, flames exist locally only where the proper balance of these three occurs. Hence the convoluted flame shape is not surprising.

It is interesting to note the pronounced effect of obstacles in the vicinity of the flame zone. The obstacles act as flame 'holders', in the sense that the flame stabilizes around them and exhibits a strong tendency to attach to them. The highest flame temperatures are observed immediately downstream of obstacles in the flame zone. In these regions, high turbulence levels exist due to the presence of recirculation zones. The high turbulence levels increase the local combustion rates, resulting in higher local temperatures. The affinity of flames to attach to objects is an important consideration for nacelle suppression studies, as fires in these locations generally are the most difficult to suppress.

The results also indicate that significant quantities of soot are generated in the nacelle fire. This observation is somewhat surprising for the problem under consideration, since the prescribed air injection rate is ~100 times that of fuel (and ~15 times the air-to-fuel ratio that is needed for stoichiometric burning). The soot generation leads to high radiative heat transfer, with peak radiative heat fluxes on the order of 100 kW/m^2. The radiative heat fluxes are highly localized near regions of high temperature *and* high soot concentration. By contrast, the peak convective heat fluxes are somewhat lower, on the order of 70 kW/m^2, and are localized near regions of high temperature *and* high velocity. Heat fluxes to clutter and surfaces are an important aspect of the nacelle fire problem, as hot surfaces represent potential points for re-ignition once suppression has occurred.

These calculations illustrate the ability of the fire field model to simulate the complex velocity and flame shapes, as well as the other larger-scale phenomena, that occur in a nacelle fire. An investigation of smaller-scale phenomena (such as detailed flame stabilization studies) and fire suppression studies utilizing the field model have recently been initiated for engine nacelle fires.

References

Holen, J., Brostrom, M., and Magnussen, B.F., 1990, "Finite Difference Calculation of Pool Fires," The Twenty-Third Symposium (Int.) on Combustion, pp. 1677-1783.

Magnussen, B.F., Hjertager, B.H., Olsen, J.G., and Bhaduri, D., 1979, "The Eddy Dissipation Concept, " The Seventeenth Symposium (Int.) on Combustion, pp. 1383-1398.

Magnussen, B.F., 1981, Particulate Carbon Formation During Combustion, Plenum Publ. Corp.

Shah, N.G., 1979, The Computation of Radiative Heat Transfer, Ph.D. Thesis, University of London, Faculty of Engineering.